ELSEVIER

CrossMark

Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 61 (2014) 1665 – 1668

Energy Procedia

The 6th International Conference on Applied Energy – ICAE2014

Thermal management and design optimization of heatsink for cooling performance improvement during transient heat generation

Mohamed Ali Rahmani[a,*], Rebei Bel Fdhila[a,c], Asa Gronqvist[b], Magnus Tysell[b], Torbjorn L Persson[b], Miroslaw Benendo[b], Zacharias Uvgard[b]

[a]ABB AB, Corporate Research, SE – 721 78, Västerås, Sweden
[b]ABB AB, SE - 721 61, Västerås, Sweden
[c] Mälardalen University, School of Business, Society & Engineering, P.O. Box 883, SE-721 23, Västerås, Sweden

Abstract

Heatsinks have long been used for cooling of electronic components to maintain them under the maximum allowed operational temperature. Forced air cooling with heatsink is suitable and enough efficient for low power applications cooling. Varieties of heatsinks are proposed by specialized industries to cool different kinds of electronic components. However, in most cases we need the appropriate heatsink to each specific case and particularly under transient heat generation that can be caused by many electronic or power electronic devices. The heat transport and evacuation process is tightly related to the heatsink performance.

This paper examines the cooling characteristics of a heatsink used in a specific industrial application. The investigation is performed using Computational Fluid Dynamics (CFD) and the heat transfer performance of the heatsink is mainly determined by the Nusselt number which can be calculated from the numerical results. Analysis and discussion of the numerical results and especially the level of Nusselt number obtained at the contact surface of the heatsink with the surrounding cooling air allow optimization of the industrial heatsink shape to meet the requested cooling performance. Comparison of cooling performance before and after heatsink design optimization showed noticeable improvement.

Peer-review under responsibility of the Organizing Committee of ICAE2014

Keywords: Heatsink, Transient heat generation, CFD, Design optimization

* Corresponding author. Tel.: +46 021345005; fax: +46 021323212.
E-mail address: mohamed-ali.rahmani@se.abb.com

Peer-review under responsibility of the Organizing Committee of ICAE2014
doi:10.1016/j.egypro.2014.12.187

1. Introduction

In this paper we only focus on the thermal performance of an industrial heatsink used with forced convection cooling to dissipate transient heat release from a semi-conductor. The semi-conductor is modeled as a heat source generating a periodic heat release such as the real components. Computational Fluid Dynamics (CFD) simulations were used first to determine the cooling performance of the actual heatsink facing such periodic heat release and especially to check if the junction temperature could be maintained under the maximum allowed temperature of 125 ºC for our semiconductor. Secondly, CFD simulations results are used to propose improvement of the whole shape design of the heatsink in order to increase its heat dissipation capacity.

Nomenclature

L	Heatsink length (m).
Nu	Nusselt number.
k_{air}	Thermal conductivity of the air (W/m.K.).
h	Convective heat transfer coefficient (W/m^{2}K.).
S_{min}	Minimum recommended fin spacing for heatsink (mm).

Two similar semiconductors are pressed between 3 heatsinks and generating the same periodic heat release of 1600 W (Fig. 1).

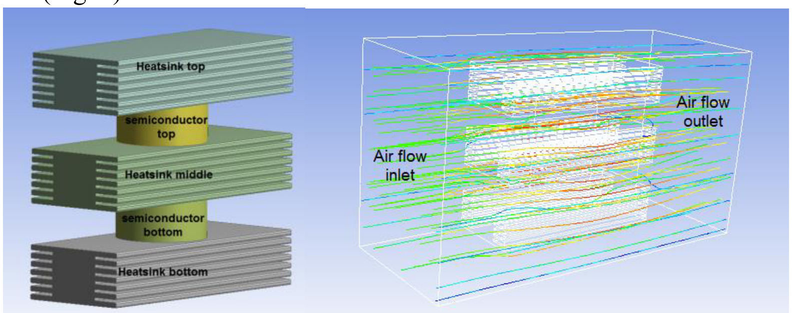

Figure 1: Schematic of assembly heatsinks and semiconductors with airflow direction

This heat is released only for 10 seconds and followed with a pause mode period of 590 seconds. The two heatsinks edges are similar but the third one in the middle has a slightly different shape.
The heatsink is a rectangular shaped aluminium extrusion with fins from both sides as shown in Figure 1. The cross-section dimension is 31x70 mm and the length is 125 mm.

2. Numerical modeling and CFD simulation

A transient 3D conjugate fluid flow and heat transfer numerical analysis is built within the CFD commercial code ANSYS Fluent. The Reynolds equations are solved together with the heat conduction using a finite volume segregated solver. Mathematical modeling of fluid flow and heat transfer equations are well described on the theory manual of ANSYS Fluent [1]. The standard k-ε turbulence model with enhanced wall treatment was used.

Physical properties of the heatsink material are those of Aluminum with a thermal conductivity of 174 W/m-K and a specific heat Cp equal to 875 j/kg-k. The periodic heat release was modeled in ANSYS Fluent with a User Defined Function (UDF).

Boundary conditions for flow fluid and heat transfer used to perform CFD simulation are detailed in the following table.

Table 1: Boundary conditions for flow dynamics and heat transfer

	Flow dynamics	Heat transfer
Inlet	Velocity inlet (V=1.598 m/s)	Constant temperature (T=25ºC)
Outlet	Pressure outlet	Back flow temperature (T=25ºC)
Walls	No slip	Thermal condition coupled

3. Design optimization

Many parameters could be used to improve design of heatsinks. In this study we focus only on the Nusselt number *Nu* and the minimum fin spacing. The Nusselt number is the ratio of convective to conductive heat transfer throw a surface. The heat transfer coefficient and the Nusselt number are related with the following relation;

$$h = Nu\ k_{air} / L \tag{1}$$

Where: L : Characteristic length, m
k_{fluid} : Thermal conductivity of the fluid,
h : Convective heat transfer coefficient

Investigation of Nusselt number plotted on the whole contact surface of the heatsink gives a visual illustration on the heat transfer performance of the heatsink.

CFD results show that heat flux and Nusselt number are higher close to the heatsink fins corner and decrease to reach zero at the fins basement. Such behavior is normal as the convective air velocity is getting lower as we approach the fin basement. But with the actual design of heatsink the zero values are more predominant even close to the fin corner which means that an important contact surface of the fins is not contributing to the heat transfer.

The actual heatsink fins have a large basement with a thickness that decreases as we approach the corner and this implies a small width of the channel between fins at the basement which is 1 mm on our case and reaches 4 mm at the corner.

The recommended fin spacing for heatsink under forced cooling mode is given by the following relation [2];

$$S_{\min} = 18.83\left(\frac{L}{V}\right)^{0.5} \qquad (mm) \tag{2}$$

Knowing the average flow velocity V (in m/s) from fans and the total length of the heatsink L (in m) we can approximate the recommended fin spacing S_{min} that should have our heatsink. S_{min} = 4 mm. Based on that value we optimized the design of the channel thickness between fins to be 4 mm on the corner and decrease slowly to 2.5 mm on the basement.

This shape modification increased the contact surface for the heatsink on the top and the bottom with the cooling air by 2% which are mainly located on the fins area. The heatsink in the middle is receiving heat from both sides and thus requires higher heat dissipation capacity. That's why besides the increase on the channel height we increased the total height of this heatsink and we designed a hole through the heatsink length parallel to the flow direction.

These modifications increased the contact surface with the cooling air by 42 % and introduced a cooling effect from inside as well. All these three modifications gave the heatsink in the middle a better heat release performance.

Comparison of hot spot temperature during each heat release cycle for both heatsink shows that the heatsink before optimization got heated over heating cycles which means that it's unable to release the cyclic heat load within the pause period of 590 seconds and induces an accumulation of heat over time (Fig. 2).

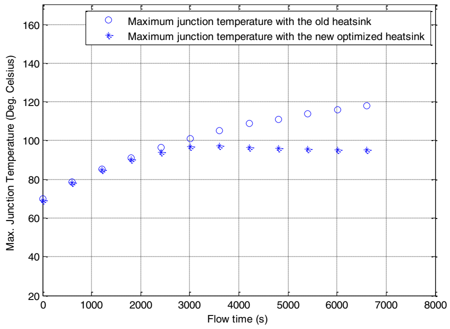

Fig. 2. Maximum junction temperature for each thermal period plotted for both CFD simulations with heatsinks design before and after shape optimization.

The heatsink with a new design succeeds after only 5 cycles to maintain and stabilize the maximum junction temperature under the maximum limit of 125 ºC (Fig. 2).

4. Conclusion

During this study a new optimized shape of an industrial heatsink is proposed. Approach to perform this optimization was to focus on the channel height between fins and introducing a new modification consisting of an ellipsoidal channel through the heatsink in the middle parallel to the air flow direction. Both modifications were sufficient to maintain the maximum hot spot temperature stable and under the allowed maximum value of 125 ºC for a specific- cyclic heat release. Deeper analysis could be a good continuation to check many others parameters on such heatsinks and adapt them to a higher heat release level and more frequent heat spikes. More design improvement can be given to optimize fin thickness, whole mass of heatsink, pressure drop and many other important parameters.

References

[1] ANSYS Fluent. Theory maual. *Mentor Graphics tutorial, Heat Sink 101*.

[2] Francois-Saint-Cyr A. Every thing you ever wanted to know about heat sinks. *Mentor Graphics tutorial, Heat Sink 101*.

Biography

Mohamed-Ali Rahmani, born in 1978 in Tunis, TUNISIA. Received his Ph.D. from the INSA of Toulouse (France) and ENIT of Tunis (Tunisia) on 2009 in Fluid dynamics and Process Engineering. Since 2009 he works at ABB Corporate Research as a researcher in Fluid dynamics and Heat transfer using experimental, modeling and numerical approaches.